MICROCOMPUTER SYSTEM FOR MEDIUM-SIZED AND EXPERIMENTAL FINITE ELEMENT ANALYSIS

Yoshiaki Yamada and Hideto Okumura
Institute of Industrial Science, University of Tokyo

Tatsumi Sakurai
Japan Advanced Numerical Analysis Inc.

SUMMARY

A microcomputer system being developed by the authors is introduced. The parallel effort of compiling a series of compact finite element analysis programs enables the execution of most computation on inexpensive microcomputers. The system is practically maintenance free and can be sustained by individual laboratories of standard scale in the educational or academic environment. As for the programs, FEMN is discussed in some detail. The program is an extended version of the original linear analysis program FEM4 and is being tested for application to problems with material nonlinearities.

INTRODUCTION

The finite element analysis has reached the stage where the execution of the structural analysis is often considered routine. This is the case particularly in the industrial environment. However, so far the execution has relied largely on expensive hardware or costly remote time-sharing services. The role of the giant main-frame or super-computer in the solution of large scale problems, e.g. the inelastic analysis of pressure vessels and piping systems operated at elevated temperature, will not be changed even in future applications. But it has been an ambition of engineers to perform a great portion of their analysis jobs on inexpensive and hopefully personal computers and thus be freed from being slave to the large systems. The development of microcomputer and associated finite element analysis programs is a breakthrough in realizing this goal.

The microcomputer system should be stand alone and almost maintenance free so that it can be sustained by individual laboratories especially in the educational or academic environment. The medium-sized engineering problems should be solved within a reasonable time limit and the system could also be adapted to multi-purpose usages, i.e. interactive compilation of fundamental computation routines, data management, preparation of engineering documents and reports, letter writing and so forth. In the present paper a compact system is introduced which is being built by the authors. In a parallel effort, a series of microcomputer finite element analysis

programs are being developed. The original version is FEM4 which is an elastic analysis program of plane stress, plane strain and axisymmetric problems (ref. 1, 2). It is extended to the nonlinear analysis program FEMN by an addition of the restart capability. The results of this innovation are manifold. By an incorporation of user defined subroutine MTRLN specifying material data, problems with material nonlinearities can be easily handled. Mesh division can be modified in the course of computation and thus the simulation and/or pursuit, for example, of crack development in fracture mechanics becomes easier. In the following, some details of the microcomputer system and the program organization of FEMN are described with the example solution of a simple pilot problem.

MICROCOMPUTER SYSTEM STRUCTURE

Figure 1 illustrates the structure of the microcomputer system almost completed at the time, May 1980, of writing this paper. Zilog Z80 is used as the 8 bit CPU (Central Processing Unit), and the capacity of main memory, which is composed of a ROM (Read Only Memory) and several RAM (Random Access Memory) boards, totals 64 kilobytes. The transfer of control, address and data between the components of the system is performed exclusively via S-100 bus. For the purpose of connection and communication a number of interfaces are installed. The 8" floppy disk drive constitutes the secondary memory for mass storage and provides the housing of a compound of operating system, supporting language, finite element analysis and other computer programs. The standard disk operating system CP/M is used so that the problems in software exchange can be avoided. At the moment, program languages are BASIC and FORTRAN. It should be noted that two 240K dual disk systems are combined for commanding four floppy disk assemblage, although a dual system kit suffices to perform the standard operation. The authors intend to shorten the overall analysis time by an adoption of parallel processing that uses several CPUs and disk drives. The contemplated inclusion of the hard disk will increase the capacity of secondary memory to a great extent and may open the way to a novel system based on 16 bit microprocessors.

Among the peripherals for I/O (Input/Output) purposes shown in figure 1, CRT unit and printer are the essentials. The function of CRT unit is manifold, as it is used for input of the data, interactive operation of the system, compilation of program segments and/or subroutines, temporary display of the computed results, preparation of documents, e.g. the users' manual, and so on. The prepared data, the completed list of the programs and the results of computation can be plugged into the printer for permanent recording. Plotter and graphics terminal are optional, but both are useful for the finite element modeling and post-processing graphics, e.g. automatic mesh generation, model editing and plotting the computed results.

The general purpose programs compiled to date for mounting on the microcomputer system are COMPOL, COMPOS, CALM, FEM4, MICRO-FEM and FEMN. The first two, written in BASIC, are essentially the microcomputer version of 'COMPOsite material computation' being developed by Tsai on a magnetic card calculator (ref.3, 4). In the program names, L and S stand for the laminate

and sandwich composite structure respectively. CALM is a matrix operation program which is basically an interactive version of the first group operations in the program CAL (ref.5, 6). FEM4 and its microcomputer version MICRO-FEM are prepared to solve the plane stress, plane strain and axisymmetric problems and then converted to FEMN to conduct nonlinear analysis. Restart capability is implemented so that the inelastic material behavior can be handled. Complementary modification is an addition of the user defined subroutine MTRLN specifying the material properties which were formerly input through element data cards. The users who tailor the subroutine MTRLN according to their material data can perform conveniently the inelastic analysis on the microcomputer. An example of MTRLN is shown in the following section.

ORGANIZATION OF NONLINEAR ANALYSIS PROGRAM FEMN

FEMN is composed of two parts FEMAB and FEMCD which are concerned with the preparation of input data file and the solution procedure of the problem. The major feature of the program is that it uses dynamic storage allocation which means the complete elimination of common statement. This function is performed by subroutines OPENS, CLOSE, PSEEK and POOLWT as shown in figures 2a and 2b.

The program organization of FEMAB is shown in figure 2a. It reads title and control cards first. Then follows the input of node and element data from which the index or integer joint array is formed and stored on IFIL2. IFIL2 accommodates also load data and IFIL6 is essentially a storage of input and processed element data. The formation of strain-displacement matrix B from the data in IFIL6 is performed by subroutine MGN and the result is written on IFIL3 to be read in FEMCD subsequently. Finally the initial displacement data (usually the cleared zero displacement) are stored on IFIL5 for subsequent updating by the solution obtained through FEMCD. In the following the principal functions of individual subroutines in FEMAB are summarized.

PINPG Preparatory segment for subroutine INPUTG
INPUTG Input generation, read input data in sequence and compile index or integer joint array
RNODE Read node data
RELEM Read element data including material specification number and element thickness
MKIDX Make index for attributing merging point and coordinate to input and processed element data and also create index for skyline assemblage of stiffness matrix
RLOAD Read loading step sequence and nodal force and/or displacement data

OPENS Open storage area for array from bottom of POOL in main memory
CLOSE Close a part or whole storage area by deleting unused array
PSEEK Search for array data by its name
POOLWT Debug write wanted array data in POOL for inspection

PMGN Preparatory segment for subroutine MGN
MGN Command sequential generation of element matrices
ISOBMN Generate element strain-nodal displacement matrix B for 3-8 variable

node parametric element

STDMA Evaluate components of strain-nodal displacement matrix B and determinant of associated Jacobi matrix J

DISPI Initialize displacements, by either clearing for initial run or entering displacement data of preceding computation for restart run

After the execution of FEMAB described above, the next program block FEMCD is called by the main of FEMN. Functions of FEMCD, whose organization is shown in figure 2b, are the formation and assemblage of the element stiffness matrix and subsequently the solution of the problem. Material data are input via the user defined subroutine MTRLN and then the stress-strain matrix D for each incremental stage of loading is evaluated in subroutine DMXMKN. FEMCD starts its operation by a transfer or reading of the data stored on IFIL5, which are title of the problem, integer data, initial cleared displacement and load data or their values obtained in preceding step of loading sequence. The stress-nodal displacement matrix S and element stiffness matrix K are formed in subroutine SMXMKN, the latter being stored in the appropriate locations in the overall stiffness matrix by referring to the index prepared on IFIL2. IFIL3 and 4 are used as the seesaw external memory for integer data and the element strain-displacement matrix B. Skyline or profile active column method of data acquisition is used for saving area in the main memory. Therefore all subroutines prefixed by capitals SK in SOLVEN, SKDCNP etc., take advantage of the skylined form of storage for the manipulation of data. Newton-Raphson iteration procedure is incorporated in subroutine SOLVEN, some details of which are discussed in the next section concerned with the solution of an elementary sample problem. The following summarises the function of individual subroutines relevant to FEMCD.

PSOLVN Preparatory segment for subroutine SOLVEN

SOLVEN Solve overall stiffness equation for unknown nodal displacement and compute reaction at constrained node; iterative procedure is incorporated in this solver for nonlinear problems

VECTWN Print out computed displacement and reaction vector at each stage of loading

SMXMKN Evaluate stress-nodal displacement matrix S and element stiffness matrix K, synthesize overall stiffness matrix by referring to merging point index stored on IFIL2, and also determine equivalent nodal force from current stress data for equilibrium check

SWRITE Write components for debugging of active columns in matrix S stored by skyline method

BWRITE Write components of vector B for debugging

SKDCNP Cholesky decomposition of symmetric positive definite matrix by skyline method

SKXMLU Multiply, add and/or subtract matrix components in skyline storage

CONVCK Check convergence of solution being obtained by Newton-Raphson iterative procedure

SKFWD Forward elimination by skyline method

SKBKW Backward substitution by skyline method

DMXMKN Evaluate components of stress-strain matrix D of constitutive equation

MTRLN User defined subroutine specifying elastic and inelastic material properties

STRSUM Add stress/strain increments to update values of stress/strain
PRINST Compute principal stresses and their directions
STRPRN Print out stress/strain solutions at respective Gauss integration points, together with coordinates of Gauss points

Finite element used in FEM4 as well as in FEMN is 4-8 variable nodes parametric quadrilateral with the following interpolation functions (ref.7).

For corner nodes 1-4

$$N_1 = S_1 - (N_8+N_5)/2, \quad N_2 = S_2 - (N_5+N_6)/2$$

$$N_3 = S_3 - (N_6+N_7)/2, \quad N_4 = S_4 - (N_7+N_8)/2$$

for midedge nodes 5-8

$$N_5 = S_5, \quad N_6 = S_6, \quad N_7 = S_7, \quad N_8 = S_8$$

where S_i stands for the trial functions defined as (ref.8, 9)

$$S_1 = (1-\xi)(1-\eta)/4, \quad S_2 = (1+\xi)(1-\eta)/4$$

$$S_3 = (1+\xi)(1+\eta)/4, \quad S_4 = (1-\xi)(1+\eta)/4$$

$$S_5 = (1-\xi^2)(1-\eta)/2, \quad S_6 = (1+\xi)(1-\eta^2)/2$$

$$S_7 = (1-\xi^2)(1+\eta)/2, \quad S_8 = (1-\xi)(1-\eta^2)/2$$

By coalescing an edge of the quadrilateral to a single point a triangular element is produced. It can be shown (ref.8, 9) that the resulting element coincides with the conventional constant stress/strain element when the primary quadrilateral element is four-noded. The number of integration points in Gauss quadrature can be one to five by five in accordance with the users' specification.

The input card or data sequence in FEMN is summarized in table I. An example of input data preparation as well as the user defined subroutine MTRLN is illustrated in the next section.

SOLUTION OF SAMPLE NONLINEAR PROBLEM AND REMARKS

As a sample problem, nonlinear behavior of a composite block specimen shown in figure 3 is analysed. The block consists of an ideally plastic element 101 and a nonlinear one 501 with a negative slope segment at a large strain as depicted in figure 4 based on the material data of figure 3. Loading sequences are summarized in figure 3 and the solid curve in figure 4 is the theoretical load-displacement relation of the block under axial tensile loading. It is noted that the loading condition in numerical analysis is given by the force increment for step 1-3 and 7-9, while in step 4-6 it is given by the displacement increment.

Table II is the image of input cards prepared for the solution of the sample problem and serves to illustrate simplicity of the data preparation.

Specifically table II is concerned with the first loading sequence, i.e. step 1-3. The solution for consecutive loading conditions, step 4-6 and 7-9 in the present example, is obtained by restarting the execution with the renewal of input data and the use of the solution obtained in the preceding step and stored on an appropriate file. Table III is the subroutine MTRLN written for this sample problem. The program FEMN is versatile because the user can easily tailor the subroutine MTRNL so that it characterizes particular nonlinear properties of the material of interest. It must be emphasized that the anisotropic material behaviors are easily incorporated in the program.

Figure 5 depicts solution convergence in the sample problem. The iterative procedure that the present version of FEMN employs is a modified Newton-Raphson method with incorporation of equivalent nodal force $\{F^a\}$. It compensates the imbalance of force equilibrium at the nodes and is given by

$$\{F^a\} = \{F\} - \int [B]^T \{\sigma\} dV$$

where $\{F\}$ denotes the prescribed nodal force, $[B]$ and $\{\sigma\}$ are the strain-nodal displacement matrix and the current stress. Convergence is satisfactory in the present example and it should be noticed that the computed results lie on the theoretical curve exhibiting sharp turning points.

Test of convergence in case of the large scale problem, sophistication of iterative procedure and extension of the program to three dimensions are the next steps that are to be taken. Moreover, the development of parallel processing and the installation of suitable hard disk will increase the speed and capacity of the system.

REFERENCES

1. Yamada, Y., Hirakawa, T., Nishiguchi, I. and Okumura, H.: Nonlinear Analysis by Finite Elements and a Microcomputer System Development, Comp. Appl. in Civil Engng., Nem Chand and Bros, Roorkee, India.
2. FEM4 Users Manual, Nonlinear Analysis Program Research Association, c/o JANA Inc., 1-1-71 Nakamegro, Megro-ku, Tokyo, 153 Japan.
3. Tsai, S.W. and Hahn, H.T.: Introduction to Composite Materials, vol.1, AFML-TR-78-201, 1978 Wright-Patterson AFB, Ohio.
4. Tsai, S.W. and Hahn, H.T.: TI-59 Magnetic Card Calculator Solutions to Composite Materials Formulas, AFML-TR-4040, 1979, Wright-Patterson AFB, Ohio.
5. Wilson, E.L.: CAL-A Computer Analysis Language for Teaching Structural Analysis, Computers and Structures, vol.10, 1979, pp.127-132.
6. Wilson, E.L.: CAL 78, User Information Manual, Rep. no. UC SESM 79-1, Univ. of California, Berkeley.
7. Bathe, K.J. and Wilson, E.L.: Numerical Methods in Finite Element Analysis, Prentice-Hall, Inc., Englewood Cliffs, 1976.
8. Yamada, Y., Ezawa, Y., Nishiguchi, I. and Okabe, M.: Reconsiderations on Singularity or Crack Tip Elements, Int. J. Num. Meth. Engng., vol.14, 1979, pp.1525-1544.
9. Yamada, Y.: Matrix Method of Mechanics of Materials, Baifukan, 1980.

TABLE I INPUT DATA FORMAT OF FEMN

```
(1) TITLE CARD (18A4)
COL  1-72    PROBLEM IDENTIFICATION ETC BY ALPHANUMERIC CHARACTER

(2) CONTROL CARD (23I1)
COL  1    =0  AXISYMMETRIC
          =1  PLANE STRAIN
          =2  PLANE STRESS
     2        NUMBER OF INTEGRATION POINTS (1-5) FOR GAUSSIAN QUADRATURE
     3*   =0  INITIAL START
          >0  RESTART
     4- 5*    NUMBER OF ITERATION IN NEWTION-RAPHSON METHOD
     6-20     BLANK
    21    >0  DEBUG WRITE IN MODULE INPUG
    22    >0  DEBUG WRITE IN MODULE MG
    23    >0  DEBUG WRITE IN MODULE SOLVEN

(3) NODE HEADER CARD (A4)
COL  1-4      'NODE'

(4) NODE DATA CARDS (A4, I6, 2F10.0, 10X, F10.0, 10X, 2I1)
COL  1- 4     'NODE'
     7-10     NODE NUMBER
    11-20     X(R) COORDINATE
    21-30     Y(Z) COORDINATE
    41-50     OBLIQUE ANGLE (DEG) OF LOCAL COORDINATE
    61    =1  X-DOF CONSTRAINED OR X(R)-DISPL GIVEN
          =0  OR BLANK  FREE OR X(R)-LOAD GIVEN
    62    =1  Y-DOF CONSTRAINED OR Y(Z)-DISPL GIVEN
          =0  OR BLANK  FREE OR Y(Z)-LOAD GIVEN

(5) ELEM HEADER CARD (A4)
COL  1- 4     'FLEM'

(6) ELEM DATA CARDS (A4, I6, 8I5, I5*, 2I5*, F8.0)
COL  1- 4     'ELEM'
     7-10     ELEMENT NUMBER
    11-15     1ST NODE NO.
    16-20     2ND NODE NO.
    21-25     3RD NODE NO.
    26-30     4TH NODE NO.
    31-35     5TH NODE NO.
    36-40     6TH NODE NO.
    41-45     7TH NODE NO.
    46-50     8TH NODE NO.
    51-55*    MATERIAL SPECIFICATION NUMBER
    56-65*    FOR EXTENSION OF PROGRAM BY USERS
    66-73     ELEMENT THICKNESS
```

(7) LOAD HEADER CARD (A4)
COL 1- 4   'LOAD'

(8) LOAD OR DISPLACEMENT STEP CARD (A4, I6, 6F10.0)*
COL 1- 4   'STEP'
     5-10   STEP NUMBER
    11-70   FOR PROGRAM EXTENSION

(9) LOAD OR DISPLACEMENT DATA CARD (A4, I6, 2F10.0)
COL 1- 4   'LOAD'
     5-10   NODE NUMBER ON WHICH GIVEN LOAD OR DISPLACEMENT IS APPLIED
    11-20   X(R) GIVEN NODAL FORCE OR DISPLACEMENT
    21-30   Y(Z) GIVEN NODAL FORCE OR DISPLACEMENT

(10) END CARD (A4)
COL 1- 3   'END'

* INDICATES ADDITION OR MODIFICATION APPLIED TO LINEAR ANALYSIS PROGRAM FEM4 AND/OR MICRO-FEM

TABLE II   INPUT DATA IMAGE OF SAMPLE PROBLEM OF FIGURE 3

```
    2-ELEMENT NONLINEAR MODEL TEST 1980-5-16 (ITER MAX= 8)
22  8
NODE
NODE     1      0.000      0.000                                               11
NODE     2      0.000     20.000                                               10
NODE     3      0.000     40.000                                               10
NODE    11     25.000      0.000                                               01
NODE    13     25.000     40.000                                               00
NODE    21     50.000      0.000                                               01
NODE    22     50.000     20.000                                               00
NODE    23     50.000     40.000                                               00
ELEM
ELEM   101      1    21    23     3    11    22    13     2     0     0..
ELEM   501      1    21    23     3    11    22    13     2     1     0 .
LOAD                                                                   .
STEP     1                                                 .............
LOAD    21   4000.000      0.000                           .  (65)
LOAD    22  16000.000      0.000                           -->  0   6.000
LOAD    23   4000.000      0.000                                0   6.000
STEP     2
LOAD    21   2000.000      0.000
LOAD    22   8000.000      0.000
LOAD    23   2000.000      0.000
STEP     3
LOAD    21   2000.000      0.000
LOAD    22   8000.000      0.000
LOAD    23   2000.000      0.000
END
```

TABLE III EXAMPLE OF USER DEFINED SUBROUTINE MTRLN

```
      SUBROUTINE MTRLN(KK,IK,EK,ITER,ISTP)
C     ..........................................................
C     . IK(4)       MATERIAL IDENTIFICATION NUMBER             .
C     . EK(IK(9))   COMPUTED STRAIN VALUE                      .
C     . EK(IK(8))   CORRESPONDING STRESS VALUE                 .
C     . ITER=0, ROUTINE DETERMINES TANGENT MODULUS             .
C     . ITER>0, ITERATES STRESS FOR COMPUTED STRAIN VALUE      .
C     ..........................................................
      DIMENSION KK(1),IK(1),EK(1)
      MID=IK(4)
      IK8=IK(8)
      IK9=IK(9)
      IF(ITER.GT.0) GO TO 6000
C     **-**  SLOPE OF STRESS-STRAIN CURVE  **-**
      IF(MID.GT.0) GO TO 3000
C     **-**  PERFECTLY PLASTIC  **-**
      EK(2)=0.3
      IF(EK(IK9).GT.10.0E-3) GO TO 2100
      EK(1)=2.0E4
      GO TO 9000
 2100 EK(1)=0.0
      GO TO 9000
C     **-**  NONLINEAR MATERIAL  **-**
 3000 EK(2)=0.3
      IF(EK(IK9).GT.3.0E-3) GO TO 3100
      EK(1)=2.0E4
      GO TO 9000
 3100 IF(EK(IK9).GT.6.0E-3) GO TO 3200
      EK(1)=0.0
      GO TO 9000
 3200 EK(1)=-1.0E+4
      GO TO 9000
C     **-**  STRESS VALUE FOR COMPUTED STRAIN  **-**
 6000 IF(MID.GT.0) GO TO 7000
C     **-**  PERFECTLY PLASTIC  **-**
      IF(EK(IK9).GT.10.0E-3) GO TO 6200
      EK(IK8)=2.0E4*EK(IK9)
      GO TO 9000
 6200 EK(IK8)=200.0
      GO TO 9000
C     **-**  NONLINEAR MATERIAL  **-**
 7000 IF(EK(IK9).GT.3.0E-3) GO TO 7200
      EK(IK8)=2.0E4*EK(IK9)
      GO TO 9000
 7200 IF(EK(IK9).GT.6.0E-3) GO TO 7400
      EK(IK8)=60.0
      GO TO 9000
 7400 EK(IK8)=120.0-1.0E4*EK(IK9)
 9000 RETURN
      END
```

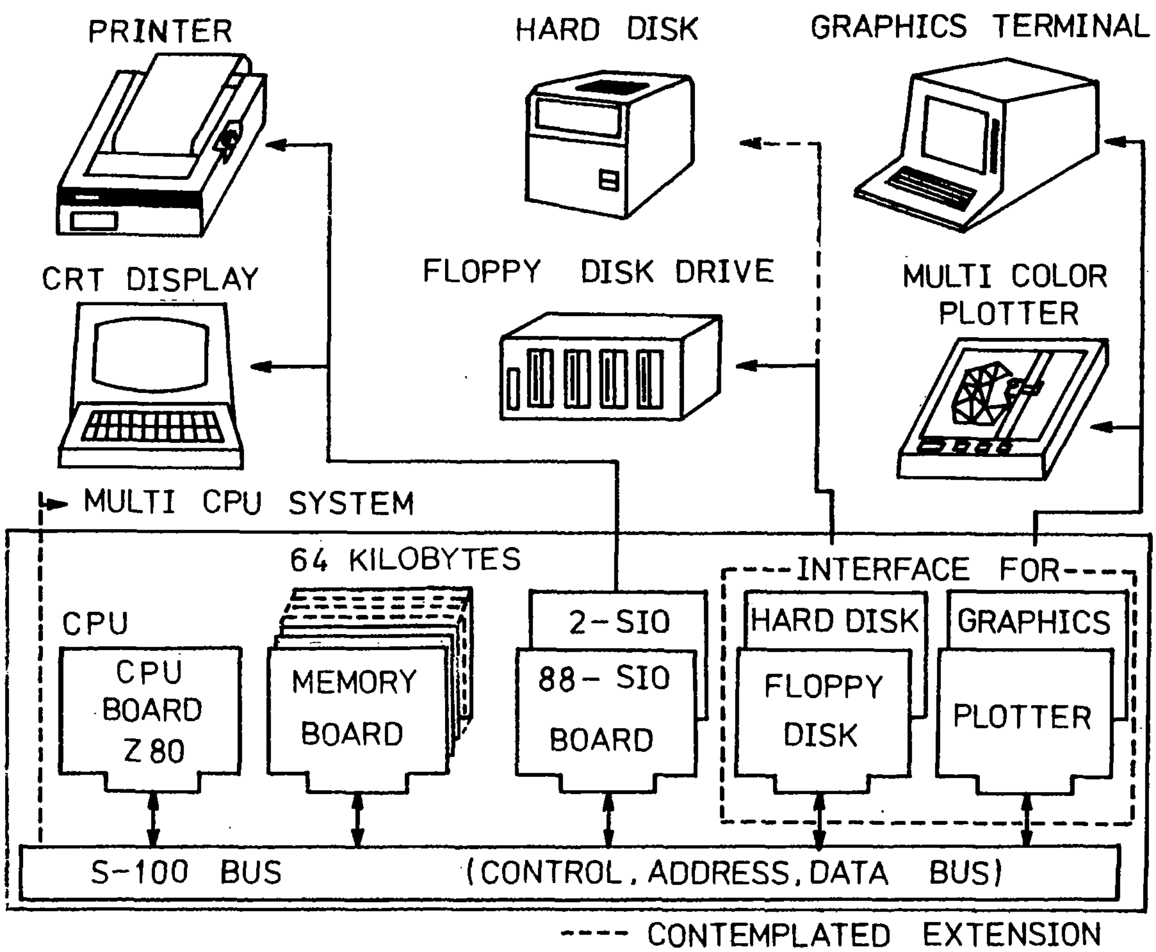
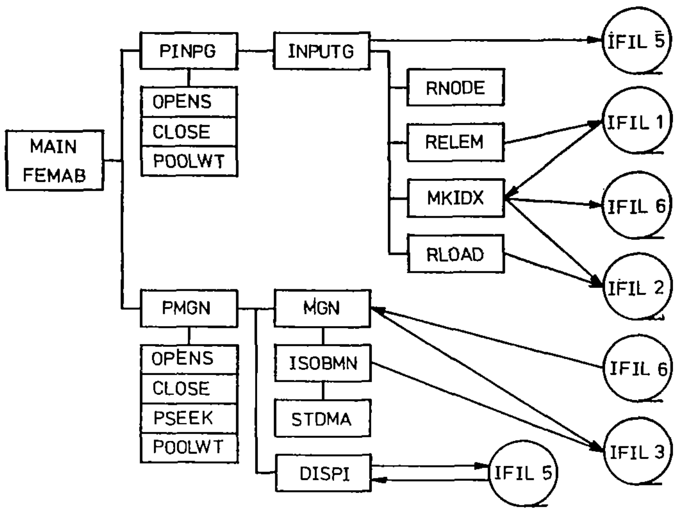
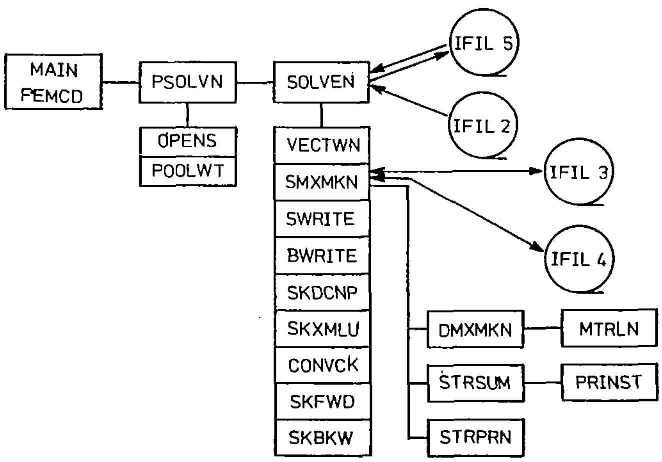


Figure 1.- Structure of microcomputer system.

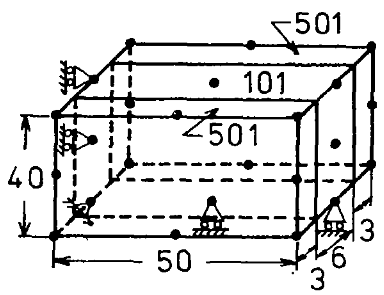


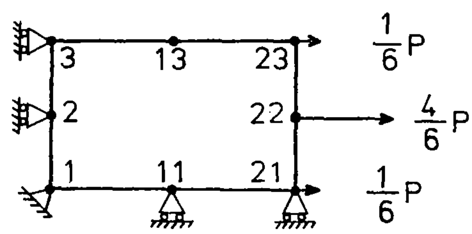
(a) FEMAB.



(b) FEMCD.

Figure 2.- Organization of FEMAB and FEMCD.





MODULUS H_0' OF MATERIALS, kgf/mm²

STRAIN RANGE	H_0' OF ELEMENT 101	501
$\varepsilon < 3\times10^{-3}$	20×10^3	20×10^3
$3\times10^{-3}<\varepsilon< 6\times10^{-3}$	20×10^3	0
$6\times10^{-3}<\varepsilon<10\times10^{-3}$	20×10^3	-10×10^3
$10\times10^{-3}<\varepsilon$	0	-10×10^3

SPECIFIED LOADING SEQUENCE

LOAD STEP	TOTAL LOAD INCREMENT	DISPLACEMENT INCREMENT
1	24×10^3 (kgf)	—
2	12×10^3	—
3	12×10^3	—
4	—	0.2 (mm)
5	—	0.2
6	—	0.2
7	-12×10^3	—
8	-12×10^3	—
9	-12×10^3	

Figure 3.- Composite block of nonlinear material.

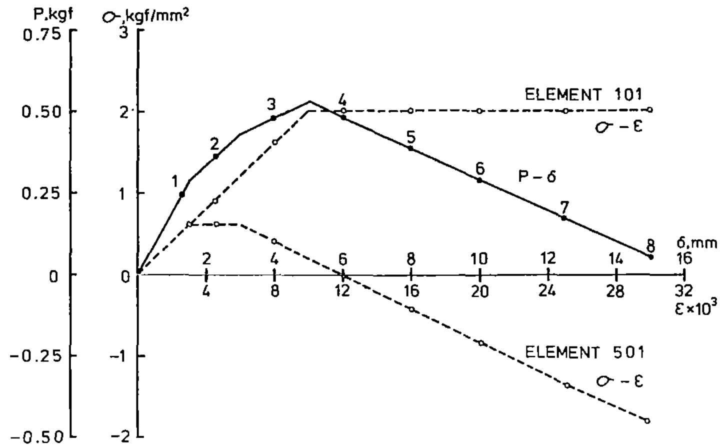


Figure 4.- Nonlinear material properties
and load-displacement curve.

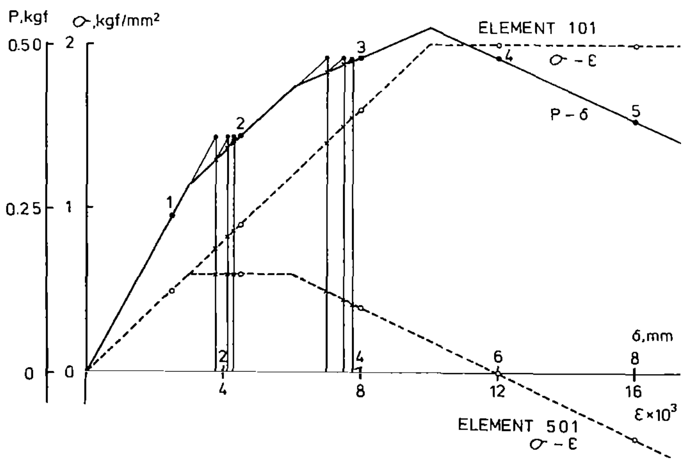


Figure 5.- Convergence of solution.